Computational Fluid Dynamics in Ventilation

theory and one-dimensional test case

Nielsen, Peter V.

Publication date:
1993

Document Version
Accepted author manuscript, peer reviewed version

Link to publication from Aalborg University

Citation for published version (APA):
Nielsen, P. V. (1993). *Computational Fluid Dynamics in Ventilation: theory and one-dimensional test case*. Dept. of Building Technology and Structural Engineering, Aalborg University. Gul serie No. 17

COMPUTATIONAL FLUID DYNAMICS IN VENTILATION

Peter V. Nielsen
Aalborg University, Denmark

2. Computational Fluid Dynamics in Ventilation - Theory and Codes

Peter V. Nielsen
Aalborg University, Denmark

THE EQUATION SYSTEM FOR BUOYANCY AFFECTED FLOW. TWO-DIMENSIONAL CASE

Transport equations.

$$\frac{\partial}{\partial x}(\rho U\phi) + \frac{\partial}{\partial y}(\rho V\phi) = \frac{\partial}{\partial x}\left(\Gamma_\phi \frac{\partial \phi}{\partial x}\right) + \frac{\partial}{\partial y}\left(\Gamma_\phi \frac{\partial \phi}{\partial y}\right) + S_\phi$$

Table 1. Values of ϕ, Γ_ϕ, S_ϕ and Buoyancy Terms

ϕ	Γ_ϕ	S_ϕ	**Buoyancy**
1	0	0 (continuity)	
U	μ_{eff}	$-\frac{\partial P}{\partial x}+\frac{\partial}{\partial x}\left(\mu_{eff}\frac{\partial U}{\partial x}\right)+\frac{\partial}{\partial y}\left(\mu_{eff}\frac{\partial V}{\partial x}\right)$	
V	μ_{eff}	$-\frac{\partial P}{\partial y}+\frac{\partial}{\partial x}\left(\mu_{eff}\frac{\partial U}{\partial y}\right)+\frac{\partial}{\partial y}\left(\mu_{eff}\frac{\partial V}{\partial y}\right)$	$-\rho\beta g\Delta T$
h	$\Gamma_{h,eff}$	0	
k	$\frac{\mu_{eff}}{\sigma_k}$	$G - \rho\varepsilon$	$+G_B$
ε	$\frac{\mu_{eff}}{\sigma_\varepsilon}$	$\frac{\varepsilon}{k}(C_1 G - C_2\rho\varepsilon)$	$+C_1\frac{\varepsilon}{k}G_B$

$$G = \mu_t\left\{2\left[\left(\frac{\partial U}{\partial x}\right)^2 + \left(\frac{\partial V}{\partial y}\right)^2\right] + \left(\frac{\partial U}{\partial y} + \frac{\partial V}{\partial x}\right)^2\right\}$$

$$G_B = \rho\beta g\,\frac{\nu_t}{\sigma_t}\,\frac{\partial T}{\partial y}$$

Turbulent viscosity, effective viscosity and effective exchange coefficient.

$$\nu_t = \frac{C_\mu k^2}{\varepsilon} \qquad \mu_{eff} = \rho(\nu + \nu_t) \qquad \Gamma_{h,eff} = \rho\left(\frac{\nu}{\sigma} + \frac{\nu_t}{\sigma_t}\right)$$

THE EQUATION SYSTEM FOR BUOYANCY AFFECTED FLOW. TWO-DIMENSIONAL CASE

Turbulent Prandtl number

$$\sigma_t = 0.67 \, \frac{1 + 0.83f + 0.112f^2 + (0.267 + 0.242f)B}{1 + 0.417f + 0.093B}$$

$$f = (k^{3/2}/\varepsilon)/c_\mu^{-3/4} \kappa d$$

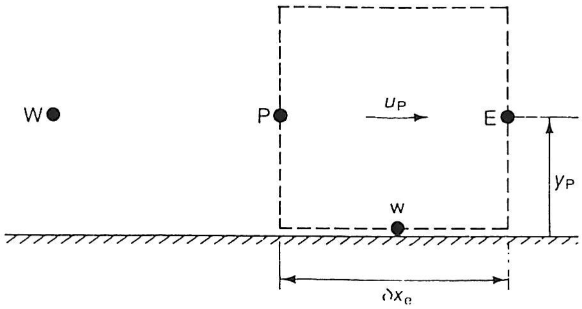
$$B = -\beta g \, \frac{k^2}{\varepsilon^2} \, \frac{\partial T}{\partial y}$$

C_1	C_2	C_μ	σ_k	σ_ε
1.44	1.92	0.09	1.0	1.3

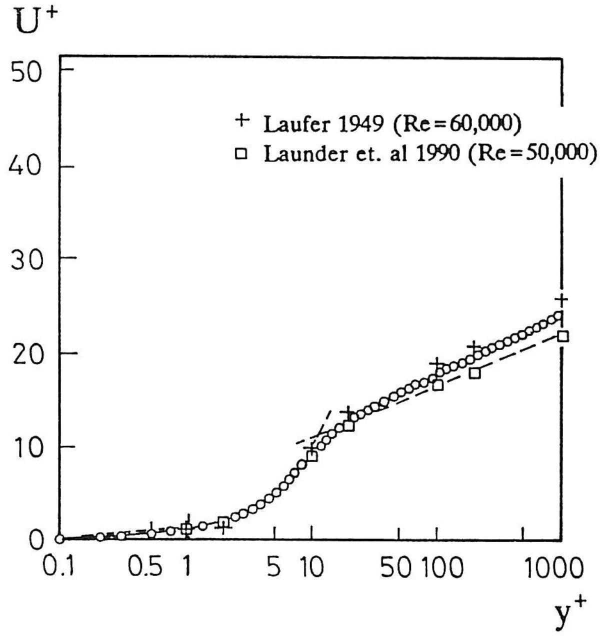
Nielsen et al.

251

Near-wall momentum control-volume.



Boundary layer flow.



$$u_P^+ = (1/\kappa)\ ln\left(E y_P^+\right)$$

where $u_P^+ = u_P/u_\tau$, $y_P^+ = u_\tau\, y_P/\nu$

$$u_\tau = \sqrt{\tau_w/\rho}$$

253

k−ε model.

$$\rho U_i \frac{\partial k}{\partial x_i} = \left(\frac{\mu_t+\mu_l}{\sigma_k}\right)\frac{\partial^2 k}{\partial x_i^2} + \underbrace{\mu_t \frac{\partial U_i}{\partial x_j}\left(\frac{\partial U_i}{\partial x_j}+\frac{\partial U_j}{\partial x_i}\right)}_{P_k} + \underbrace{\beta g_i \frac{\mu_t}{\sigma_t}\frac{\partial T}{\partial x_i}}_{G_k} - \rho\epsilon$$

$$\rho U_i \frac{\partial \epsilon}{\partial x_i} = \left(\frac{\mu_t+\mu_l}{\sigma_\epsilon}\right)\frac{\partial^2 \epsilon}{\partial x_i^2} + \frac{\epsilon}{k}(c_1 P_k + c_3 G_k - c_2 \rho\epsilon)$$

Minimum value concept.

Velocity and velocity gradients → 0.

$$\mu_t = \rho C_\mu \frac{k^2}{\epsilon}$$

k and ϵ are restricted to values in such a way that $\mu_t \rightarrow const\ \mu_l$.

Damping function f_b.

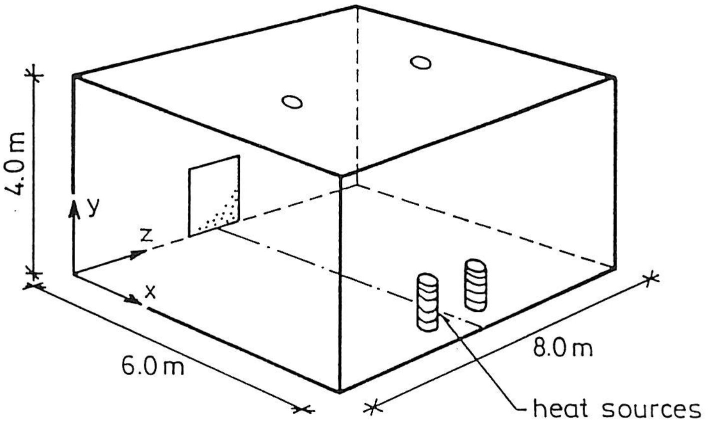
$$\mu_t = f_b \rho C_\mu \frac{k^2}{\varepsilon} \text{ where } f_b = \text{func}\left(G_k/\epsilon\right)$$

Damping function $f_{R_t} f_b$ (Chikamoto, Murakami and Kato).

$$\mu_t = f_{R_t} f_b \rho C_\mu \frac{k^2}{\epsilon}$$

$$f_{R_t} = \exp\left(\frac{-3.4}{(1+R_t/50)^2}\right) \qquad f_b = \begin{cases} 0.0 \ for \ b \le -10 \\ 1+b/10 \ for \ -10<b<0 \\ 1.0 \ for \ b \ge 0 \end{cases} \qquad R_t = \frac{\rho k^2}{\mu_l \epsilon} \qquad b = \frac{G_k}{\epsilon}$$

254

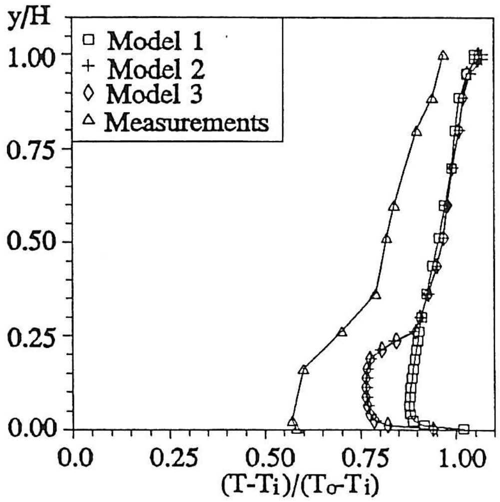


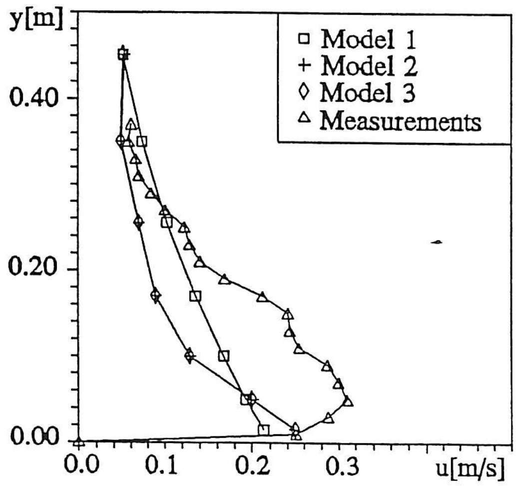
The predictions are made with wall functions, adiabatic boundaries and a radiative energy transfer model.

Model 1: Standard $k-\varepsilon$ without G_k-term

Model 2: Standard $k-\varepsilon$ with G_k-term

Model 3: $k-\varepsilon$ with damping function $f_{R_t}\ f_b$





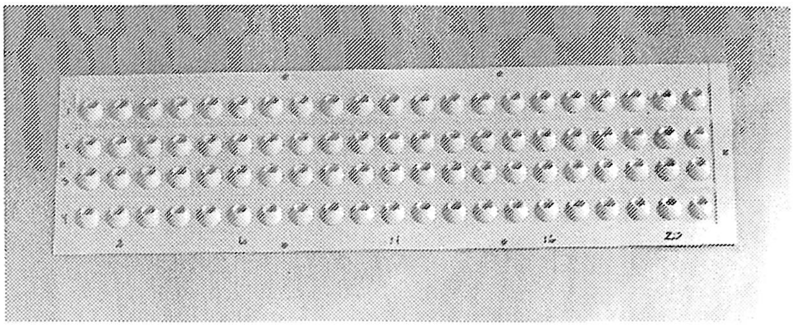
Jacobsen and Nielsen.

DIFFERENT TYPES OF BOUNDARY CONDITIONS

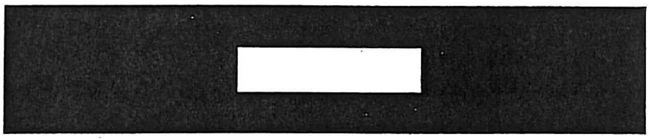
- Direct description, where the boundary conditions are located at the surface of the air terminal device.
- Simplified geometry with actual momentum flow.
- Front area method with actual momentum flow. Supply area equivalent to front area.
- Momentum method with actual momentum flow and volume flow. Supply area equivalent to front area.
- Box method, where the boundary conditions are located at the surface of a box at some distance from the air terminal device.
- Prescribed velocity method, where some of the variables are described analytically inside a volume in front of the air terminal device and the remaining variables are predicted by the numerical method.
- Computer-generated supply conditions, where the flow in the air terminal divice is studied as a first step in the predictions of flow in the whole room.

SIMPLIFIED DESCRIPTIONS OF A SUPPLY OPENING

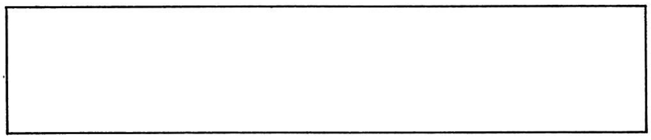
IEA air terminal device.



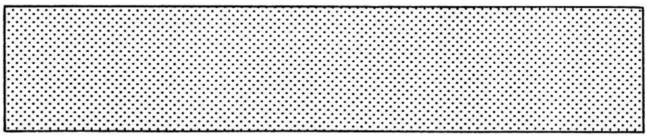
Simplified geometry.



Front area method .



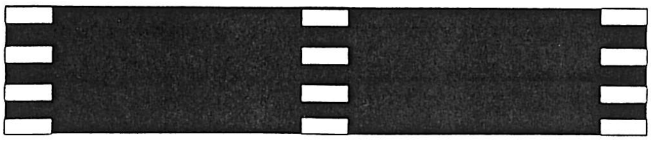
Momentum method.



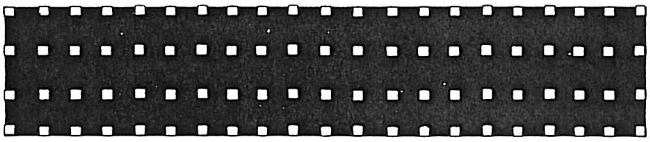
Nielsen, ASHRAE Transaction 1992

SIMPLIFIED DESCRIPTION OF A SUPPLY OPENING

12-slots method*.

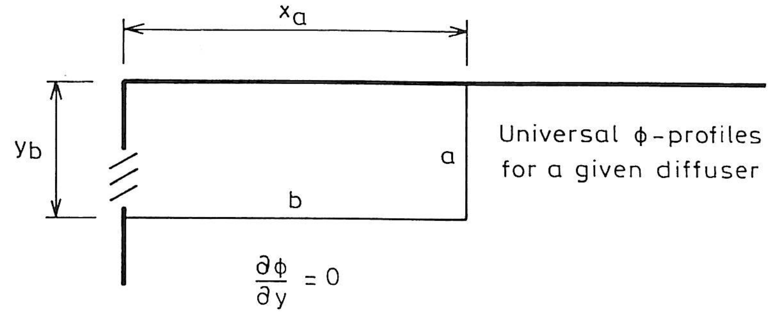


84-slots method*.

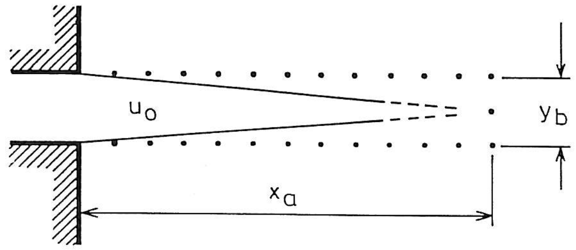


* Chen, Moser and Suter, A database for assessing indoor airflow, air quality, and draught risk.

Location of the boundary conditions in the box method.

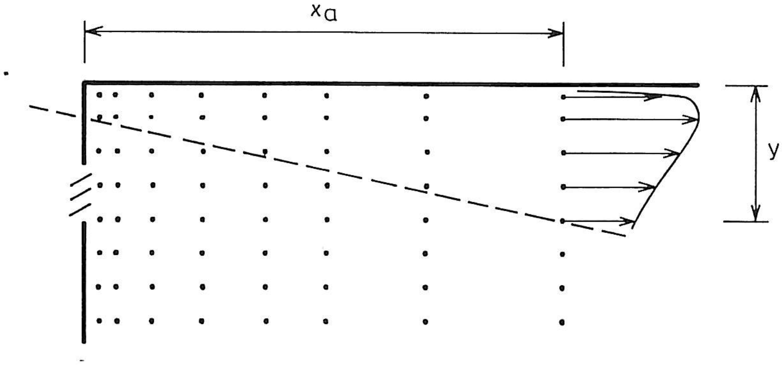


Boundary conditions located close to the constant velocity core of a jet.

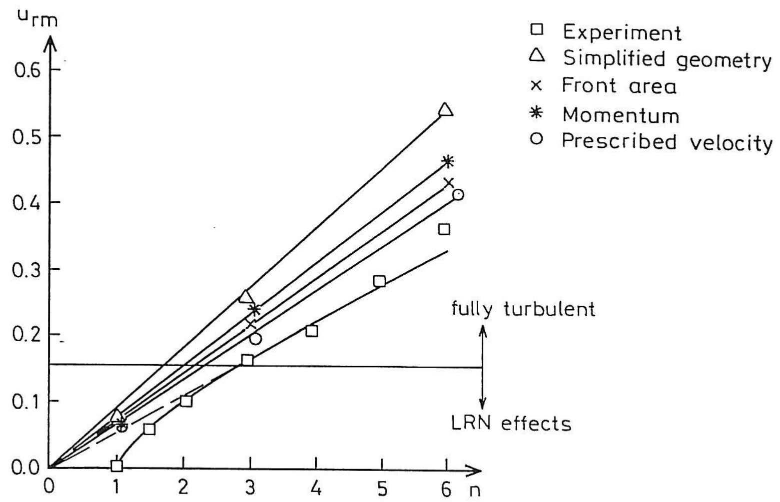


PRESCRIBED VELOCITY METHOD

Prescribed velocity field close to the supply opening.

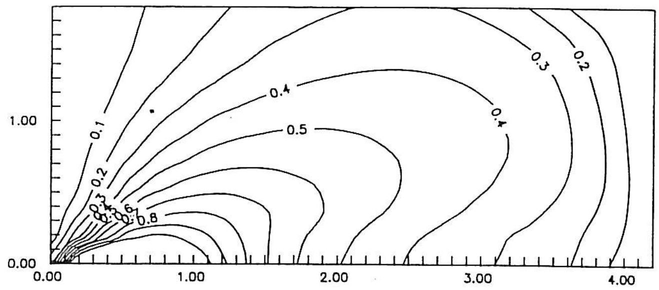


Maximum velocity in the occupied zone u_{rm} with different descriptions of the IEA diffuser.

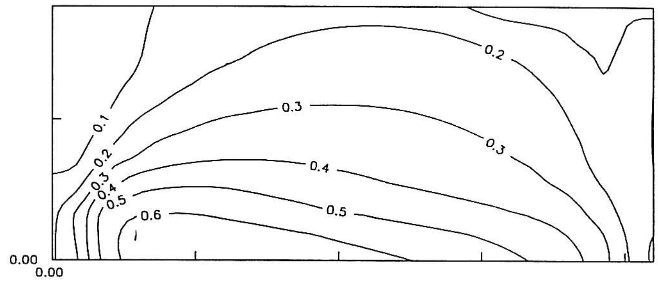


DIFFERENT DESCRIPTIONS OF SUPPLY OPENING

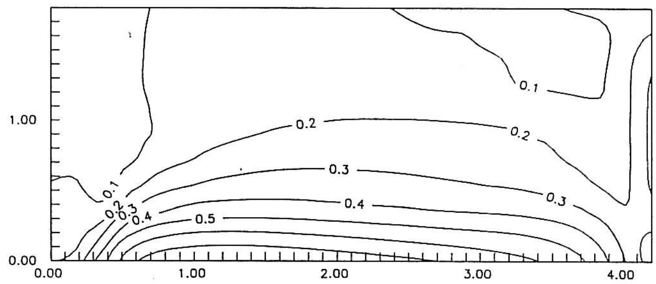
Wall jet flow at the ceiling with different descriptions of the IEA diffuser.



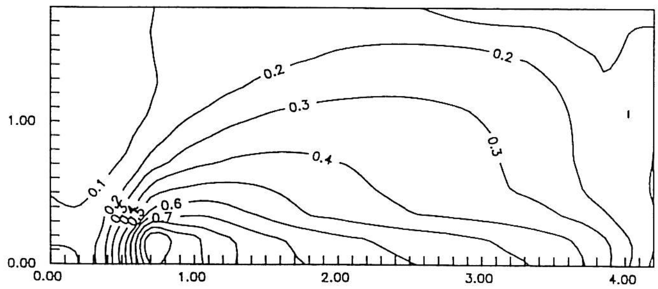
Simplified geometry.



Front area method.



Momentum method.



Prescribed velocity method.

3. Computational Fluid Dynamics in Ventilation - The IEA Annex 20 Work

Peter V. Nielsen
Aalborg University, Denmark

The basic aims of IEA are to

- Increase energy security
- Develop alternative energy sources
- Develop energy conservation

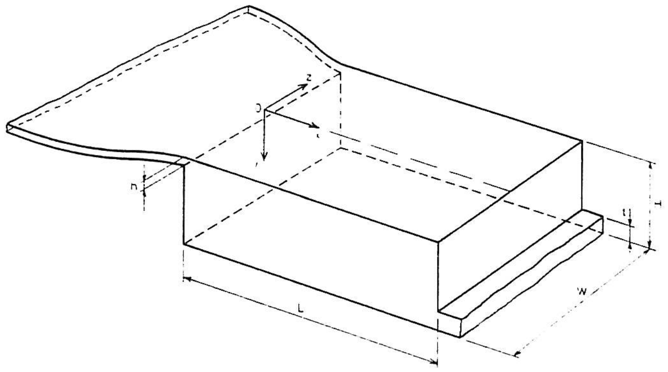
Annex 20, subtask 1

- Prediction of accurate air flow patterns within rooms to obtain energy efficiency

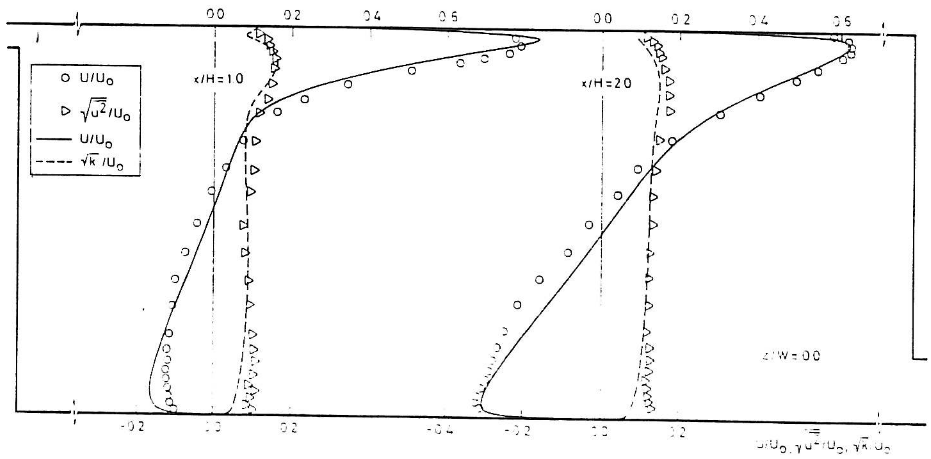
Research work in participating countries

- Comparative test of 2D CFD-codes
- Comparative test of 3D measurements in full-scale rooms
- Comparative test of 3D CFD-codes

Laser-doppler and hot-wire measurements of isothermal flow



Measurements and CFD-predictions with low Reynolds' number $k-\varepsilon$ model

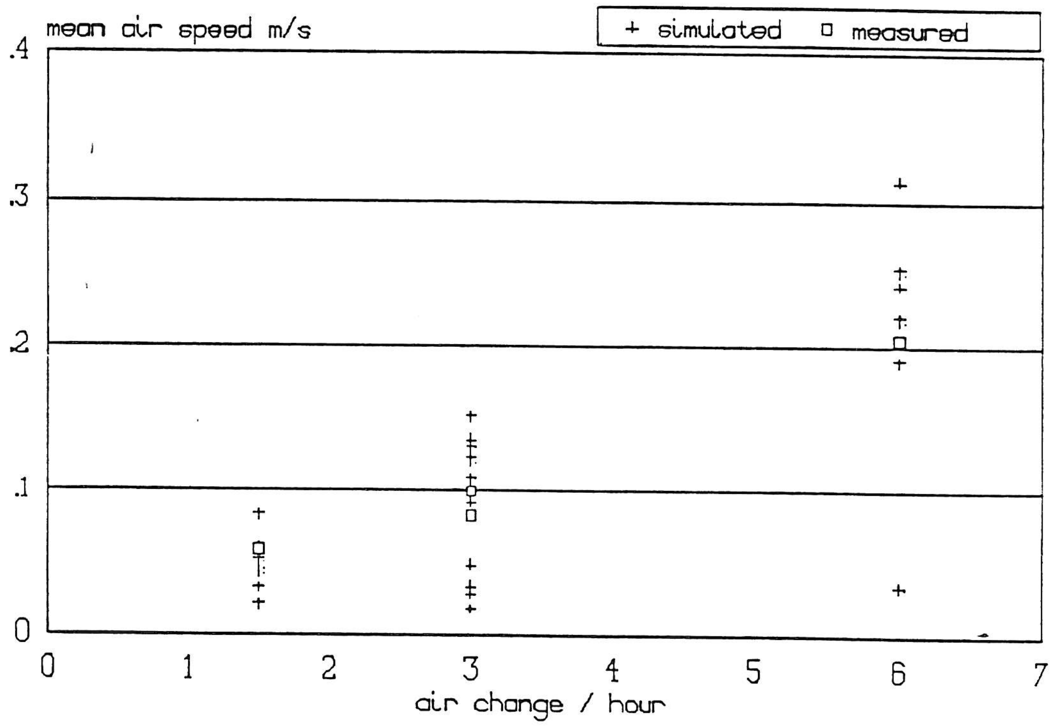


Test case B represents forced convection at isothermal conditions.

Case B 1 ($n = 1.5\ h^{-1}$) represents a low Reynolds number case. The air flow rate is around the minimum value required to ventilate an office room. The throw of the jet is about 3/4 of the room length.

Case B 2 ($n = 3\ h^{-1}$) is the basic case. The air flow is around the usual value in office rooms.

Case B 3 ($n = 6\ h^{-1}$) represents a high Reynolds number case. Velocity measurements can be made with good accuracy. Important case for comparison of measured and calculated results.



Subtask 1. Summary report.

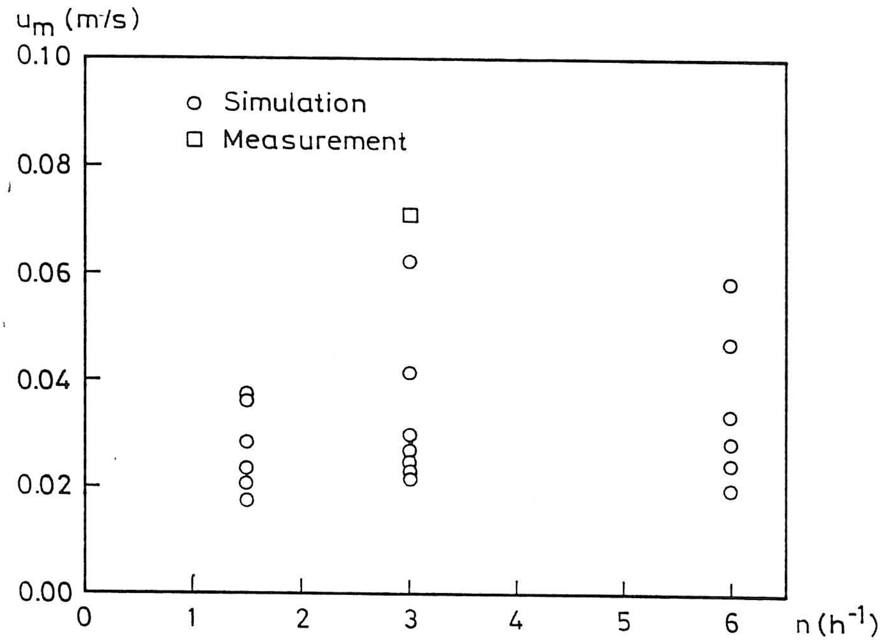
248

Test case D represents free convection with a radiator located beneath a cold window. Three different radiators and window surface temperatures.

Case D 1 (T_{rn} = 46 °C, T_{wd} = 10 °C) represents double glazing and ambient temperature of ~ -10 °C. Heat load is about 19 W/m².

Case D 2 (T_{rn} = 55 °C, T_{wd} = 5 °C) represents single glazing and ambient temperature of ~ 0 °C. Heat load is about 28 W/m².

Case D 3 (T_{rn} = 65 °C, T_{wd} = 0 °C) represents single glazing and ambient temperature of ~ -10 °C. Heat load is about 28 W/m².



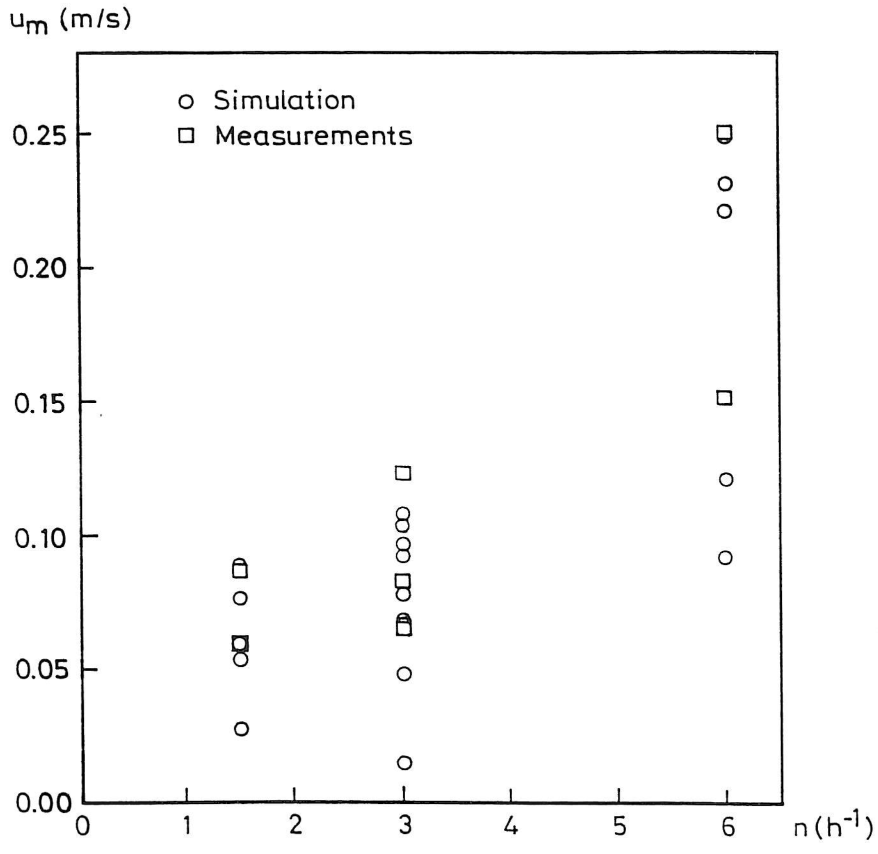
Subtask 1. Summary report.

Test case E represents mixed convection under summer cooling conditions at three different supply flow rates.

Case E 1 (n = 1.5 h^{-1} , T_o = 10 °C and T_{wd} = 30 °C) represents a high Archimedes number case with reduced penetration depth of the jet. Cooling load of 15 W/m^2.

Case E 2 (n = 3 h^{-1}, T_o = 15 °C and T_{wd} = 30 °C). A usual airflow rate. Cooling load ~ 15 W/m^2.

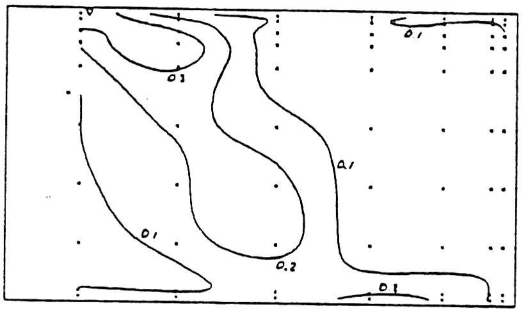
Case E 3 (n = 6 h^{-1}, T_o = 15 °C and T_{wd} = 35 °C). Low Archimedes' number case. Cooling load is 30 W/m^3 which is quite normal.



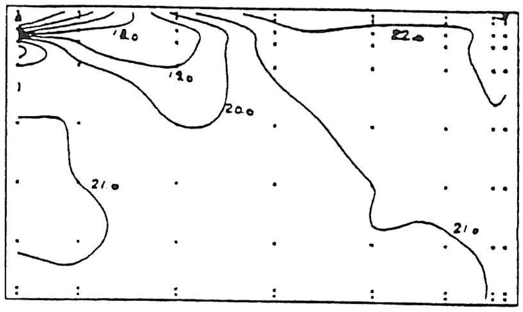
Subtask 1. Summary report.

Measurements

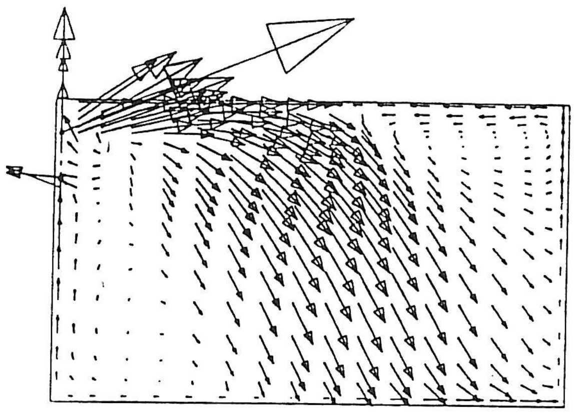
Velocity

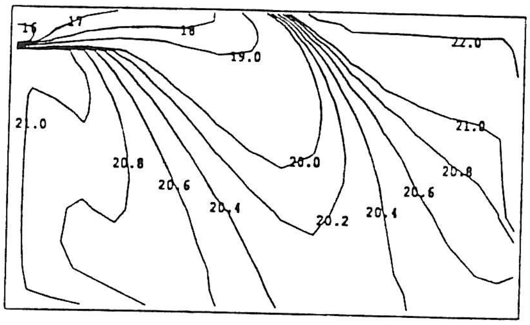


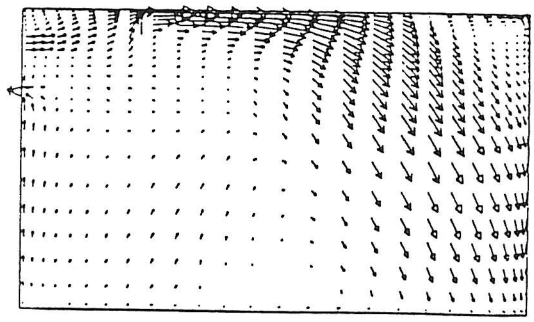
Temperature

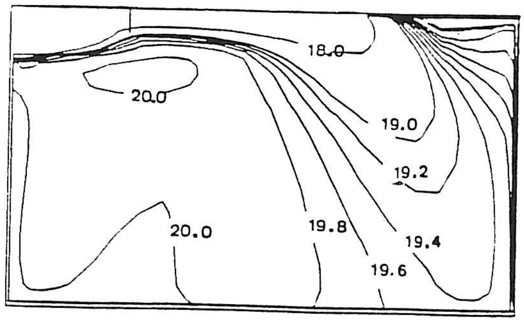


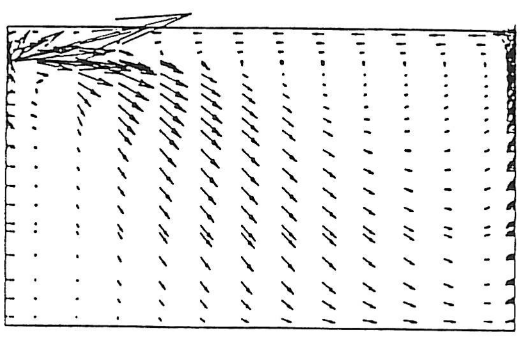
Predictions





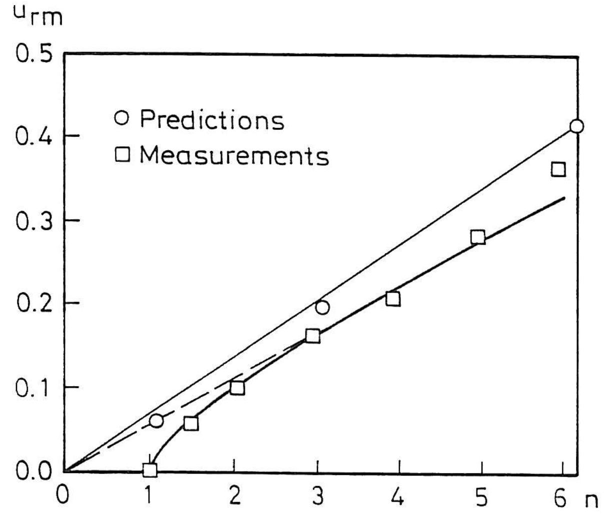






Subtask 1. Summary report.

IEA Annex 20 work. Maximum velocity in the occupied zone at isothermal flow.



4. Commercial Application of CFD in Ventilation

Peter V. Nielsen
Aalborg University, Denmark

COMMERCIAL APPLICATION OF CFD IN VENTILATION

Typical markets for CFD:

> Airport centres, large theatres, atria, shopping malls, oil platforms, inter-connected rooms in buildings, industrial buildings, etc.

Flow and comfort in small rooms (office rooms etc.) are not the typical market for CFD.

Typical problems to be solved:

> smoke movement in large constructions, energy flow, energy consumption, excess temperature control, draft, contaminant transport, ventilation effectiveness, combined free and forced convection, pressure distribution, overall picture of thermal comfort, etc.

High priority areas, acitivities and quantities:

> fast pre-processing, effective visualization software, stable numerical scheme, effective handling of complicated geometry, high computation speed, effective handling of "real life" boundary conditions.

This situtation will often have the effect that quantities and activities as

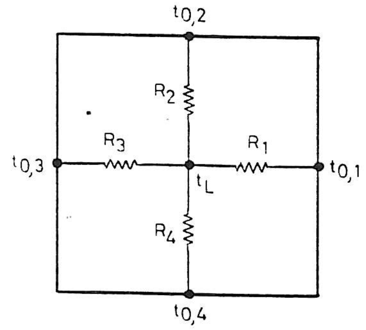
> advanced wall functions, complicated turbulent models, individual boundary conditions for air terminal devices, grid independent solutions, body fitted grid, etc.

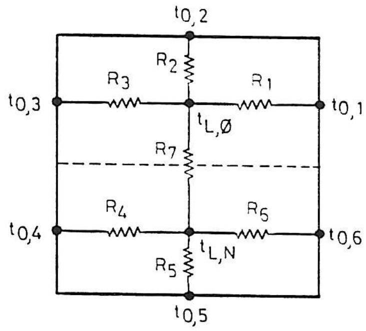
will obtain a lower priority.

The commercial application of CFD may be looked upon as an advanced **zonal model.**

The accuracy of the predictions can be compared with the accuracy of a flow element model, but the CFD method can handle situtations which are difficult with a flow element model.

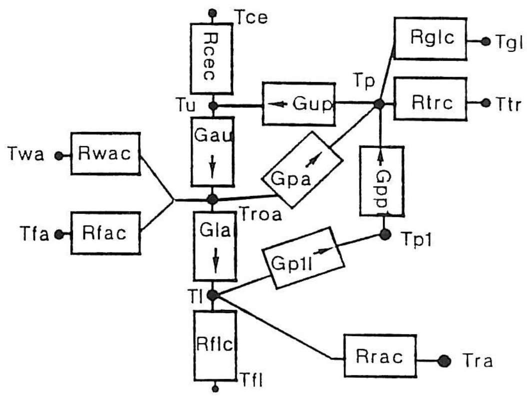
One-zone and two-zone models.

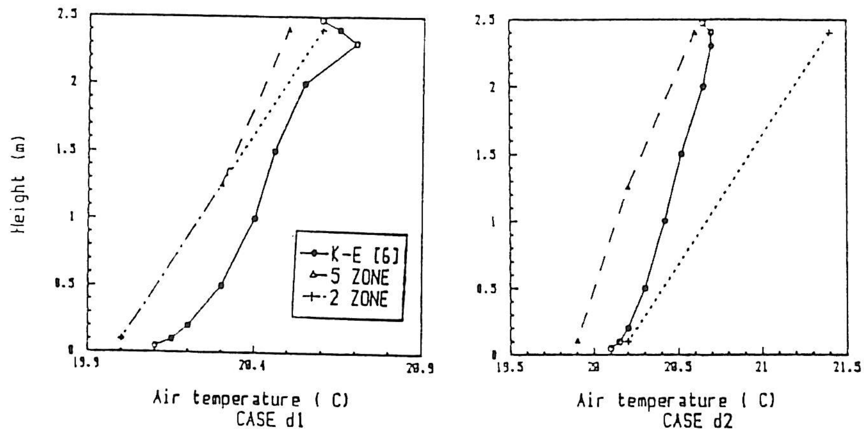




Five-zone model (IEA, Annex 20, Case D).

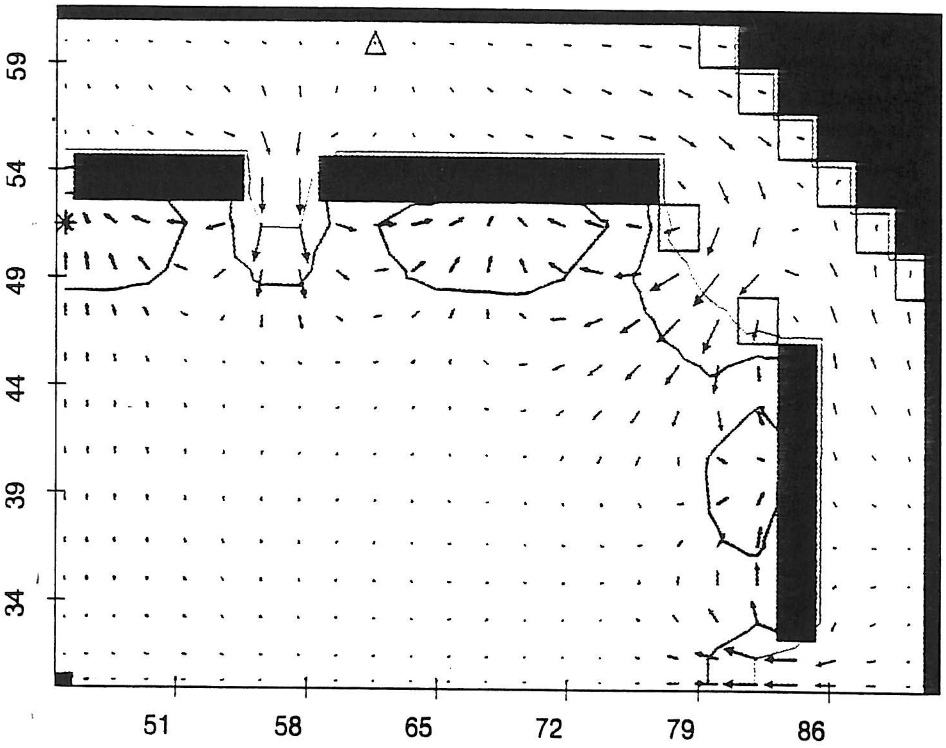
- air leaving the radiator (Tp1)
- thermal plume (Tp)
- upper zone (Tu)
- central zone (Troa)
- lower zone (Tl)





Inard and Buty.

Preliminary predictions of the air movement in Gjøvik Olympic Rock Stadium.



The numerical scheme corresponds to a zonal model concept.

- Large control-volumes (2 m × 2 m × 1 m).

- Prediction of mass flux, momentum flux and energy flux.

- All boundary conditions can be simplified. (It is only necessary to specify the flux at the control-volume surface).

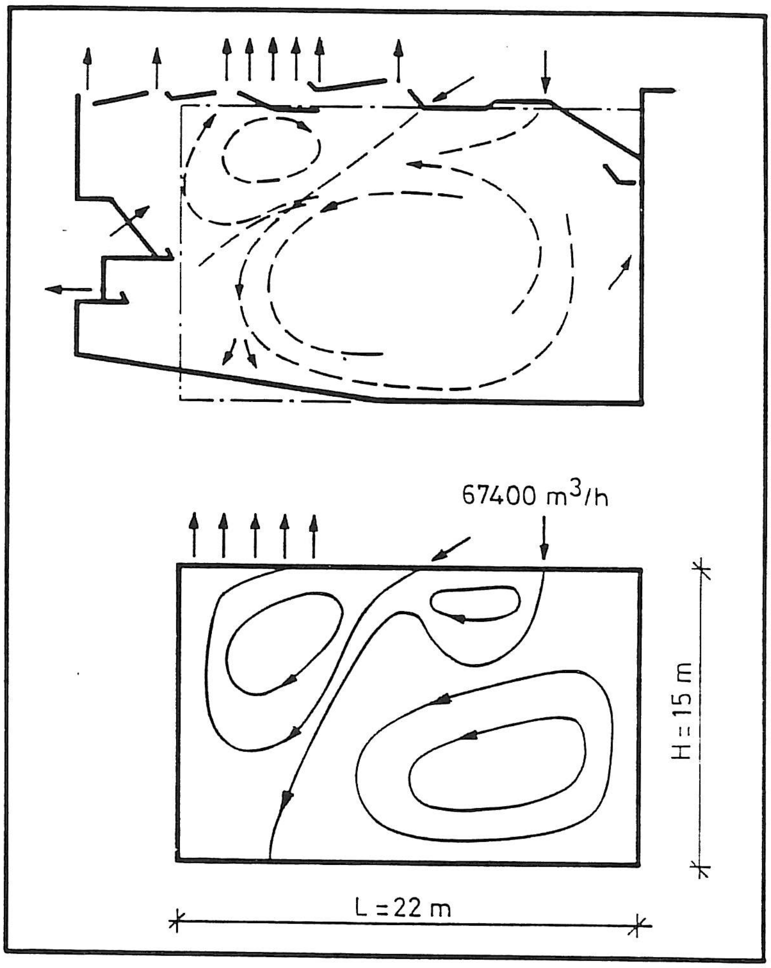
- Advanced turbulence models are not important due to reduced velocity gradients. (Numerical diffusion may also be large).

The present preliminary method is advanced compared to a zonal-model or compared to a flow element model.

CFD predictions by Mathisen and Tjelflaat.

EXAMPLE OF AIRFLOW SIMULATION IN A THEATRE

- Simplified geometry
- Simplified boundary condition at supply openings
- Low number af grid points



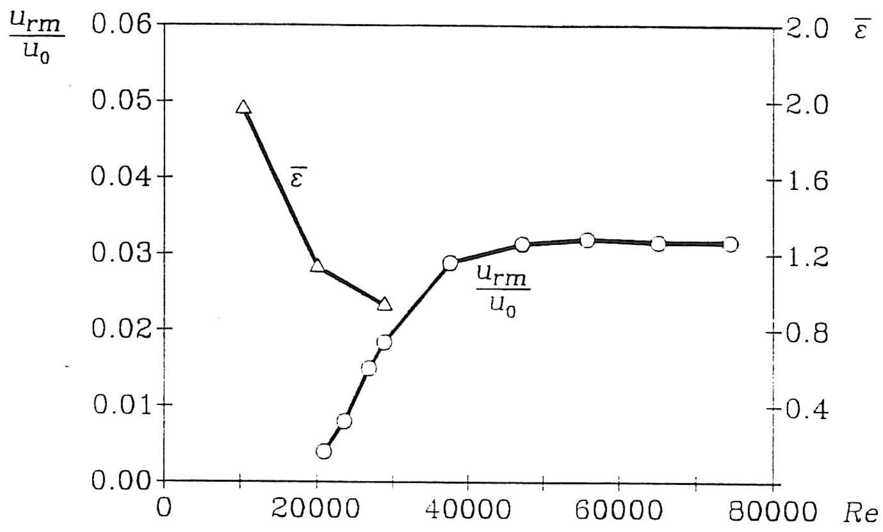
Ehle and Scholz.

239

5. New Developments in CFD with Relevance to Ventilation

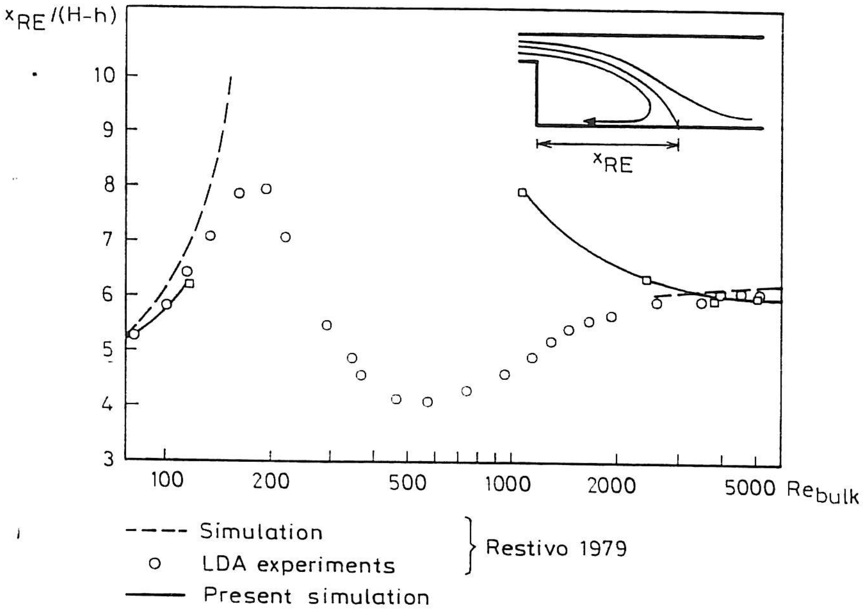
Peter V. Nielsen
Aalborg University, Denmark

Normalized maximum velocity and ventilation effectiveness in a forced ventilated room.

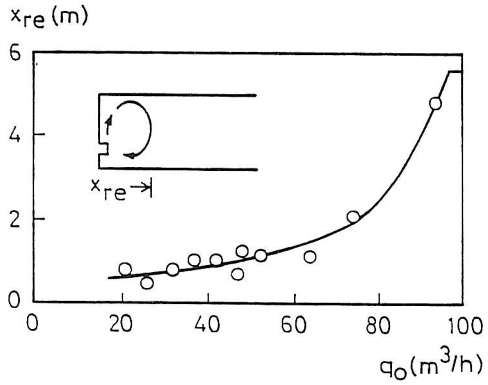


234

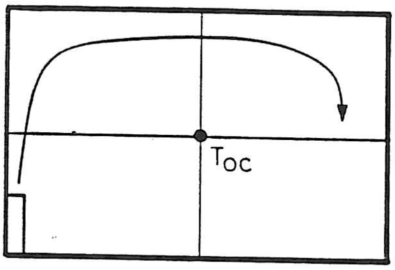
Model experiment (H = 30 mm) and predictions with a low Reynolds number k-ε model.



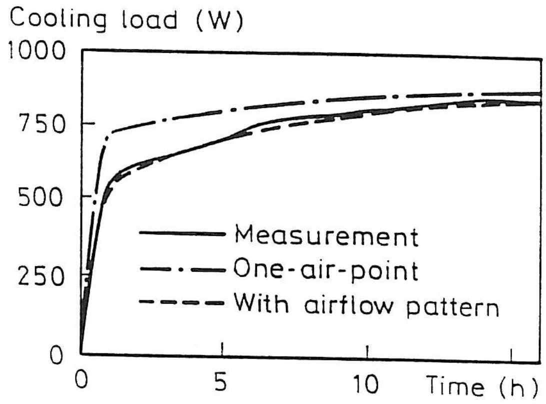
Full-scale experiments (H = 2.4 m).



Skovgaard and Nielsen.



One-air-point model



A data base with velocity distribution for different Archimedes' numbers is precalculated and used at each time step in the cooling load programme and temperature distribution is obtained from a solution of the energy equation.

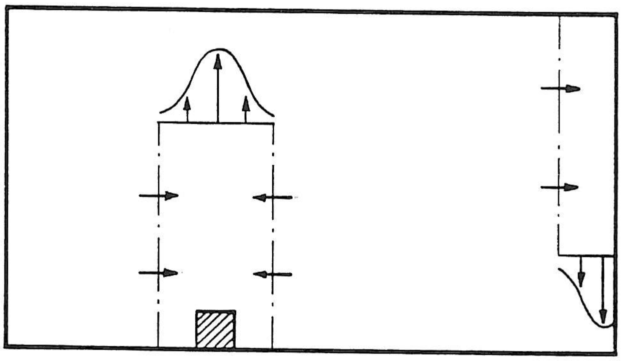
The figure shows the measured and the calculated cooling load in a room where 950 W is applied as a step function.

An one-air-point model does not take account of the vertical temperature gradient, while a cooling load model with airflow pattern is able to handle this effect and results are in this case in good agreement with measurements.

Chen and Kooi.

244

AREAS WITH ANALYTICAL DESCRIPTION OF FLOW ELEMENTS



Certain areas can be described by flow elements (thermal flow over a heat source, cold downdraft, concentration distribution around a person, etc.).

The flow can be specified by the box method or the prescribed velocity method.

It is possible to have feedback from the flow field (e.g. $\partial T/\partial y$ influence on thermal flow).

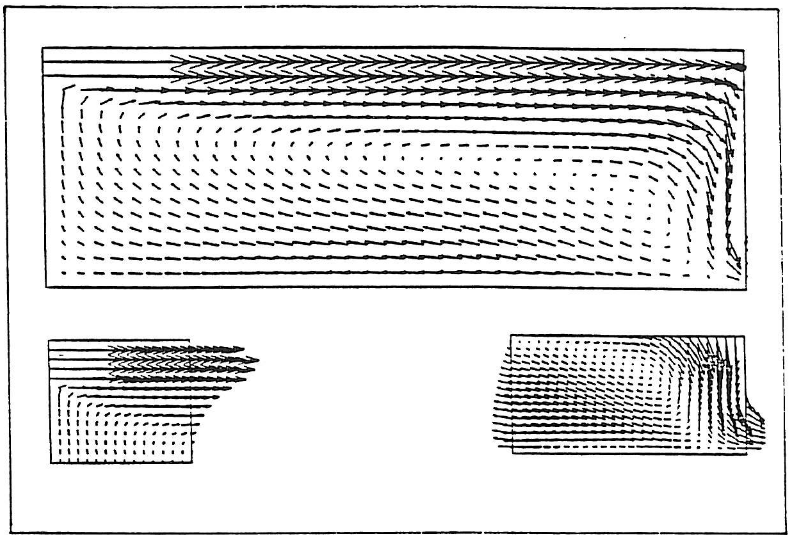
The method is a combination of CFD and a zonal model. (A zonal model is a connection of flow elements).

Advantages:

- Reduced number of grid points.
- Fast pre-processing.
- Easy use of "real life" components and measurements.

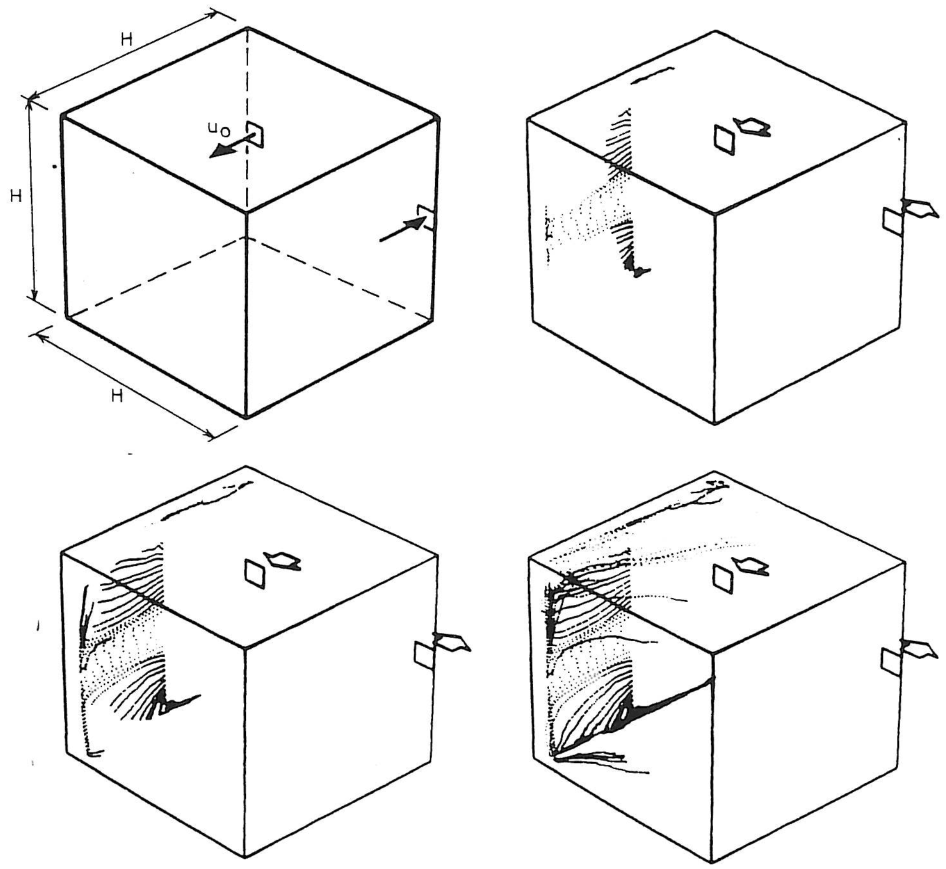
245

Grid refinement at inlet and outlet. Much faster convergence than for the corresponding single-grid method.



Li, Fuchs and Bai.

250



The LES method is based on Navier-Stokes' equations, the continuity equation and the energy equation, respectively, all filtered with respect to grid space, but not to time. The equation system is closed by an expression for subgrid scale (SGS) Reynolds' stresses. The turbulence is represented as the time dependent solution of this equation system, and it is important that the grid spacing is fine enough to allow a description of the energy containing eddies.

If mean flow quantities have to be predicted, transient calculation must be conducted over a time which is sufficient to obtain the average values.

Murakami.

242